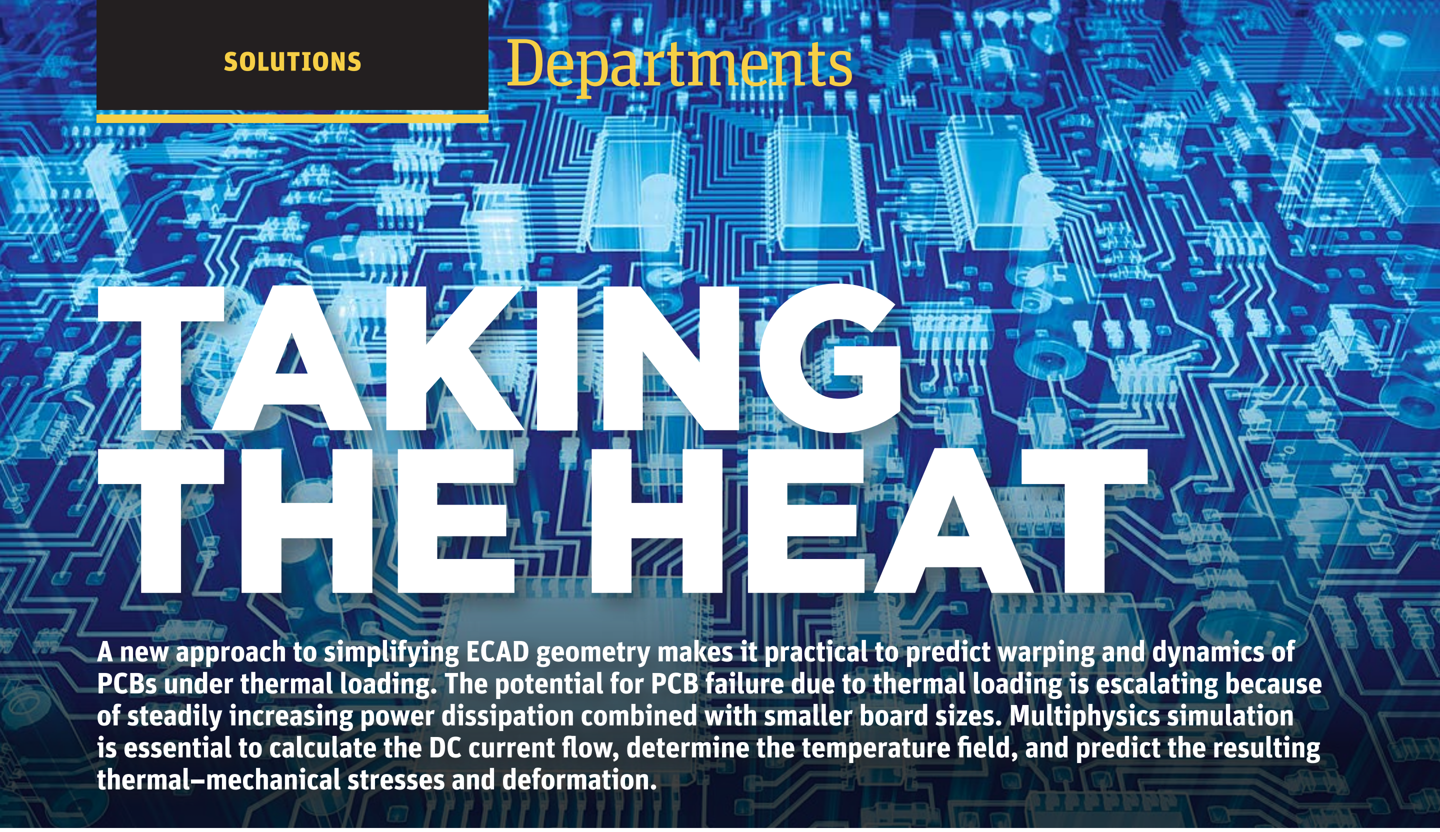


TAKING THE HEAT

A new approach to simplifying ECAD geometry makes it practical to predict warping and dynamics of PCBs under thermal loading. The potential for PCB failure due to thermal loading is escalating because of steadily increasing power dissipation combined with smaller board sizes. Multiphysics simulation is essential to calculate the DC current flow, determine the temperature field, and predict the resulting thermal–mechanical stresses and deformation.

By Tim Pawlak, Research and Development Fellow, ANSYS

Thermal stress is a constant concern in printed circuit board (PCB) design. For example, thermal cycles can generate stress in the PCB itself due to the difference in the coefficients of thermal expansion (CTE) of copper and the epoxy resin typically used in a PCB. As the temperature changes from hot to cold in repeated cycles, the solder balls that provide contact between the PCB and the integrated circuit (IC) package are stretched and squeezed. This can cause separation from the IC package or the PCB, leading to disastrous electrical failure. The difference in the CTE between the ceramic package and the PCB may generate additional board deformation.

Damage due to electronics failure because of thermal cycling is not just a nuisance: It can be catastrophic. The report on why an AirAsia flight crashed in December 2014, killing 162 passengers, concluded that crew member errors when attempting to recover from a rudder malfunction due to a cracked solder joint was the cause of the disaster.

In the past, it had not been practical to accurately simulate the amount of thermal deformation of a proposed board design. Engineers have been able to determine point sources of heat from manufacturer's specifications and use ANSYS SIwave to calculate Joule heating in PCB traces and vias. They could then use these heat sources as inputs for an ANSYS Icepak simulation, and determine temperature fields on and around the board. Furthermore, they could apply these temperature fields to an ANSYS Mechanical simulation of the board to predict its deformation under manufacturing and operating cycles. However, accurately determining the mechanical properties of the complete board geometry, including all traces and vias, was not possible. The resulting model was so complex and large that it could not be solved within the time frame of a normal design iteration.

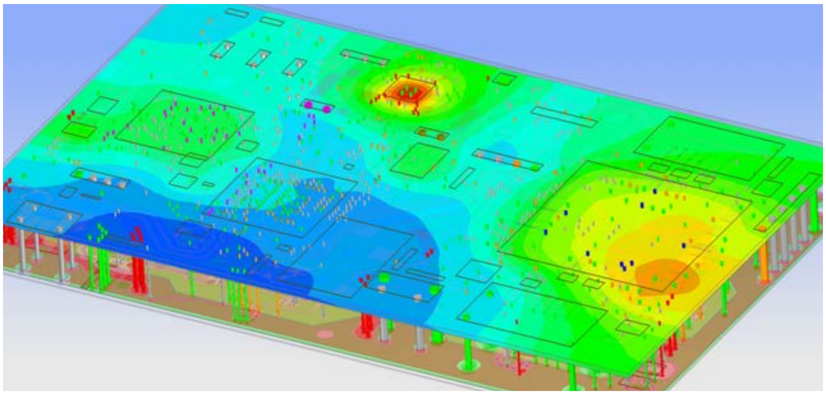
ANSYS has overcome this obstacle with a new multiphysics methodology that simplifies PCB geometry while accurately tracking its material properties at any point. This method effectively simulates board performance under thermal loading.

SPEED UP YOUR TIME FOR ANALYSIS WITH ANSYS SPACECLAIM — WEBINAR
ansys.com/heat101

By understanding the effects of thermal loading on the structural integrity and reliability of the board, engineers are empowered to design products with lower failure rates and reduced warranty costs while also reducing time to market and engineering expenses.

COMPUTING DC SOLUTION

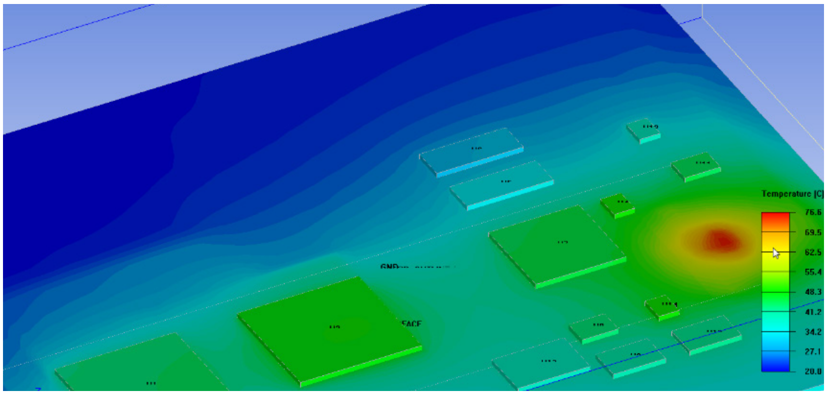
The first step is to use SIwave to compute DC currents and voltages throughout the PCB. SIwave then calculates the current density throughout the board, which in turn determines Joule heating (the process by which heat is produced due to electrical resistance when an electrical current flows through a conductor). Joule heating has increasingly become important as a source of thermal loading in PCBs as board sizes are reduced and power consumption remains steady or rises.



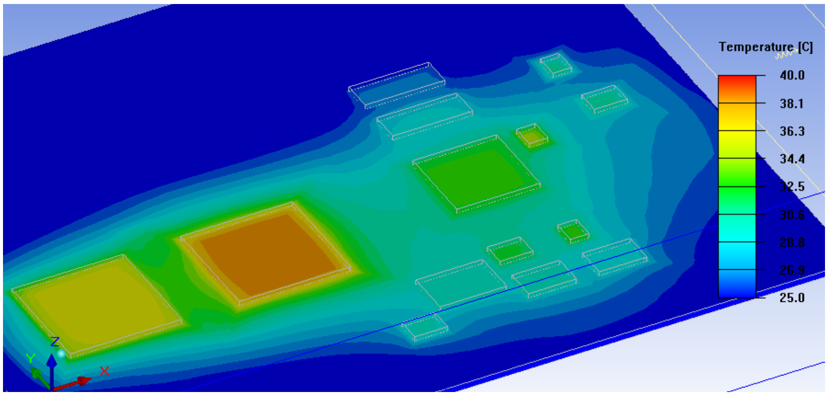
ANSYS SIwave generates a DC solution.

PERFORMING THERMAL SIMULATION

With the release of ANSYS 17.0, ANSYS SIwave and ANSYS Icepak bidirectional workflow has been automated, making it much easier to determine Joule heating. This capability imports the board, trace map and current density predictions, and sets the thermal boundary conditions for Icepak, which uses the trace map to calculate the orthotropic thermal conductivity of the PCB. This is important because much of the heat generated on the board is dissipated via convection or radiation from the board itself. Icepak solves fluid flow equations and includes all modes of heat transfer — conduction, convection and radiation — to compute temperatures at every point in the solution domain. The macro then exports the resulting temperatures from Icepak back into SIwave, which updates the electrical properties for the DC solution based on the temperature field. SIwave then recalculates the DC field and exports it to Icepak. This iterative process continues until the power dissipation and temperature results have converged.

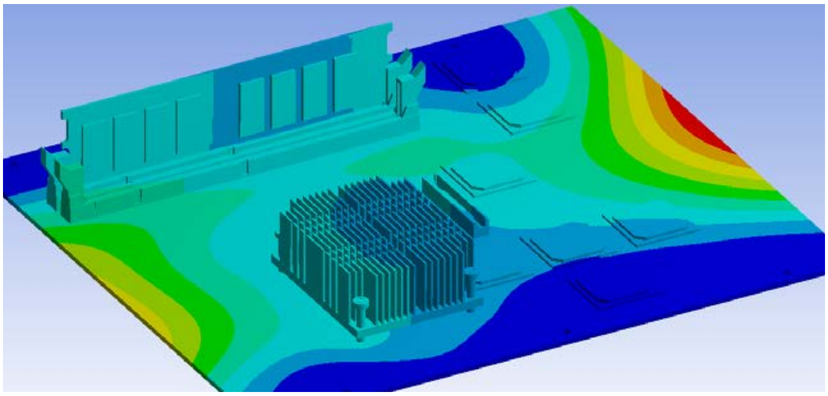


ANSYS Icepak predicts maximum temperature with Joule heating at 76.6 C.



ANSYS Icepak predicts maximum temperature without Joule heating at 40.0 C.

A new multiphysics methodology simplifies PCB geometry while accurately tracking its material properties at any point.

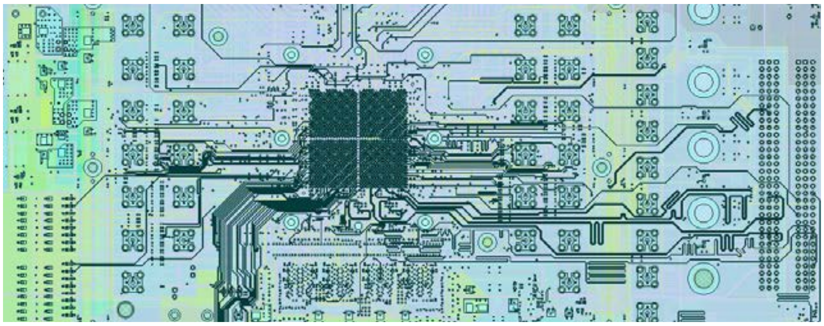


ANSYS Mechanical predicts response under random vibration loading.

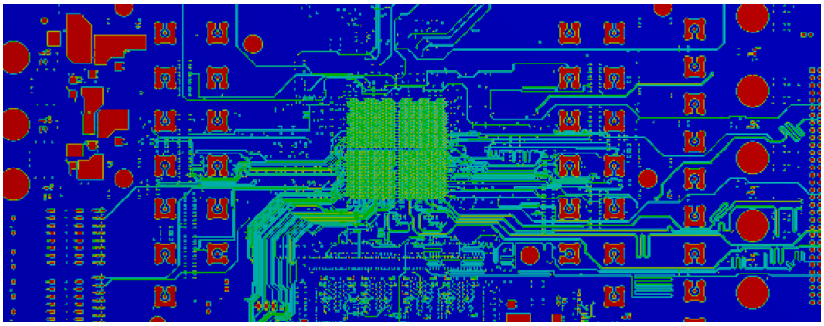
This methodology gives engineers the ability — for the first time — to accurately determine the effects of thermal loading on a PCB within a timeframe that is relevant to a typical design cycle.

CALCULATING TRACE METAL FRACTION

The next step is to build the structural model for computation of deformation, strain and stress. ANSYS SpaceClaim reads the ECAD geometry and converts it into a 3-D model of simplified layers. That geometry is then opened in Mechanical, where it is easily meshed due to the layer nature of the geometry. Each layer can have a combination of metal and dielectric. The metal of the ECAD model is mapped onto the mesh on an element — wise basis. Some elements will be entirely metal, others entirely dielectric, and still others a mixture of metal and dielectric. A higher mesh density will produce a more accurate representation of the model and take more computational time. Lower mesh density will take less computational time and be less accurate.



Full geometry in ANSYS SpaceClaim



ANSYS Mechanical model showing trace metal fractions.

ANSYS Mechanical predicts deformation on the board with Joule heating.

CALCULATING THERMAL-MECHANICAL STRESSES AND DEFORMATION

The resulting model with its mesh is the basis of a structural simulation with appropriate material properties taken into consideration. Its solution will quantify thermal stresses, strains and deformation at any location on the board. By knowing these locations, engineers are able to determine whether some aspects of the model are at risk of failure, including attachment locations such as at solder balls. The engineer can also perform dynamic analysis to determine the modal frequencies and effects of random vibration on the board with thermal loading taken into account.

This multiphysics simulation methodology gives engineers the ability — for the first time — to accurately determine the effects of thermal loading on a PCB within a time frame that is relevant to a typical design cycle. Engineers can now identify thermal loading issues, propose possible solutions and determine whether or not each of their proposed solutions solves the problem long before a prototype is produced. By understanding the effects of thermal loading on the structural integrity and reliability of the board in the early stages of the design process, engineers are empowered to design products that have lower failure rates and reduced warranty costs while also reducing time to market and engineering expenses.

Thermal stress can cause solder balls to separate from the IC package or the PCB, leading to disastrous electrical failure.